



Tutorial 2g.02

Consolidation under unbankment

Ref: CESAR-TUT(2g.02)-v2025.0.1-EN

1. INTRODUCTION

1.1. Tutorial objectives

The problem considered in this tutorial is the construction of an embankment on a multi-layered saturated soil. The aim of the analysis is to obtain the evolution of the settlements over time due to soil consolidation. Two types of analysis are performed:

- Short-term (undrained) and long-term (drained) settlement calculation,
- A fully coupled consolidation (CSNL module, consolidation of material with nonlinear mechanical behaviour).

1.2. Problems description

An embankment lies on a saturated soil consisting of 3 horizontal layers. We intend to observe the long-term soil settlement due to the weight of the embankment. For this reason, the embankment is modelled by a trapezoidal distributed pressure in downward direction with respect to the variation of embankment thickness. Both the geometry and the loading of embankment are symmetric, so that the model can be reduced to one half of system. Moreover, taking into account the length of the embankment compared to its area section, the model can be analysed in Plane Strain.

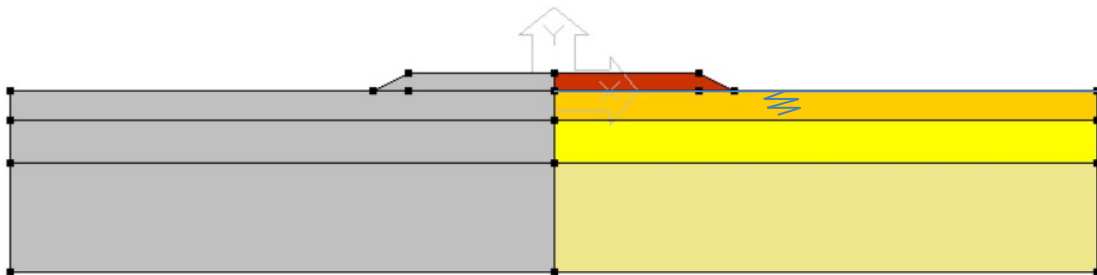


Figure 1: simplification of the problem geometry

Simplified geometry:

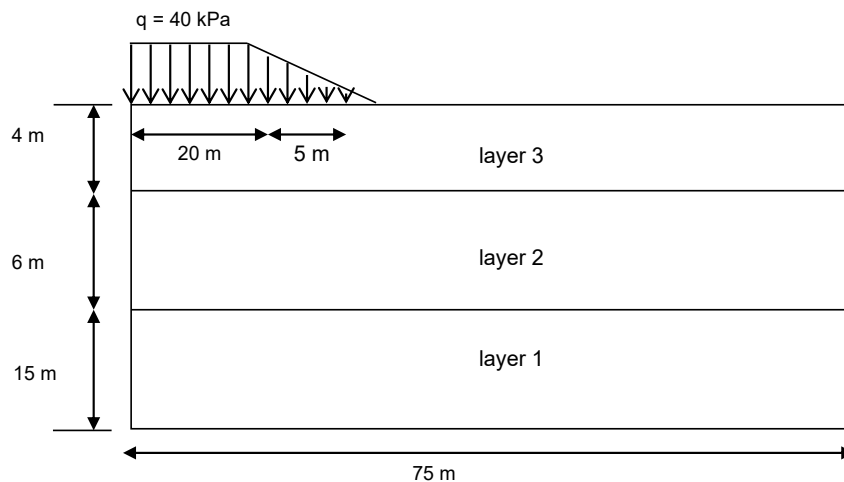


Figure 2: 2D Plane strain model description

Boundary conditions:

1. On the symmetrical axis : Horizontal displacement is null.
2. At the base : Vertical displacement is null.
3. At the lateral extension : Horizontal displacement is null.

Loadings:

It is preferable to avoid instantaneous loadings in a coupled calculation and in an elastoplastic analysis. This is because the loadings may introduce a discontinuity of the fields of solutions with respect to time so that they are prejudicial to the stability and the precision of the numerical schemes. Actually, the majority of the loadings taken into account in geotechnics are applied over an interval of time more or less important: one thus recommends applying the loadings in a progressive way.

In this tutorial study, we recommend to take into account a progressive increase in the applied loading with time for an interval $[0, T]$ as indicated in Figure 2.

The embankment will be modelled as a volume. Its weight will be increased with time (increments).

Note that, as alternative, the embankment could also be modelled by a non-uniform pressure of 40 kN/m^3 .

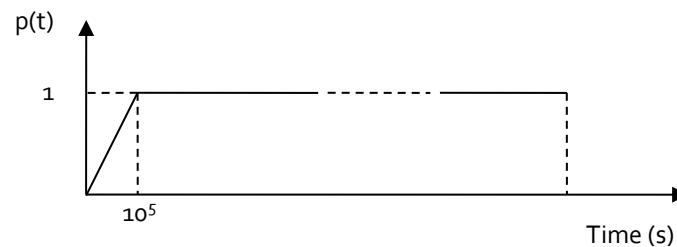


Figure 3: Increments of load with time



Note that, as alternative, the embankment could also be modelled as a volume and the associated self-weight as loading.

2. GEOMETRY AND MESH

2.1. Geometry

A new project always starts in the tab **GEOMETRY**.

Drawing of the geometry:

Use the **Points** dialog box  to create the geometry as shown in Figure 1.

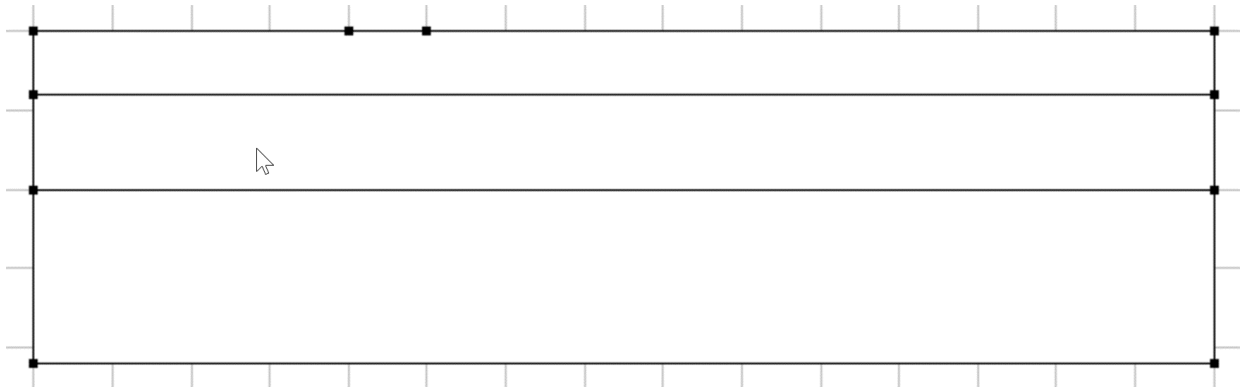


Figure 4: Contours of the model

2.2. 2D Meshing



Refine the mesh density in the area of high strains, i.e. below the embankment. Use a progressive density definition to generate a progressive evolution from small segments to large segments on the boundary edges.

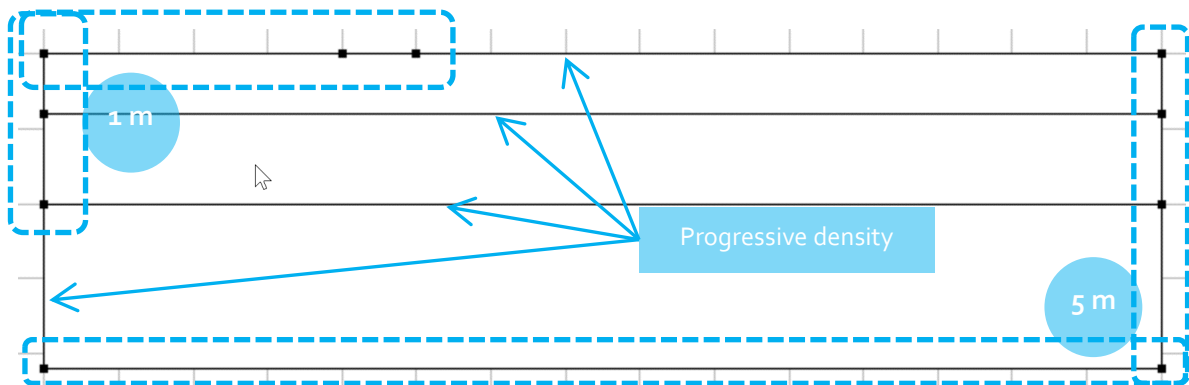


Figure 5: Mesh density plan

Mesh density definition:

1. Go to the **MESH** tab on the project flow bar to start the definition of divisions along lines.
2. Select all the contours of the embankment (as pictured on Figure 5)
3. Click on  **Fixed length density** to impose a length for the divisions of these segments. Enter **1 m** in the dialog box. Click on **Apply**.

4. Select the external contours of the model.
5. Click on  **Fixed length density** to impose a length for the divisions of this segment. Enter **5 m** in the dialog box. Click on **Apply**.
6. Click on  **Progressive density** to divide the segment with a variation of lengths. Tick **First division** and **Last division** to define the method. Enter **1 m** as **First division** and **5 m** as **Last division**. Click on the segments between the embankment and external edges

 The position of the click defines where the initial division is. The software algorithm will adjust the length for the best fit with the input value of length.

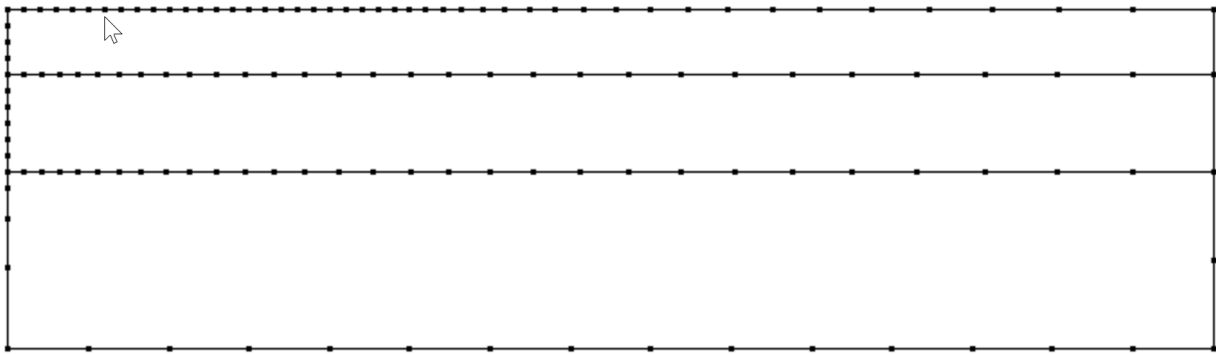


Figure 6: Mesh density

Mesh generation:

1. Click on the **Surface meshing** tool .
2. Select all the surface bodies of the model.
3. Choose **Quadratic** as interpolation type. Choose **Triangle** as element shape.
4. Click on **Apply** to generate the mesh.

 CESAR-LCPC proposes 3 levels for the surface meshing procedure, giving the possibility to generate a coarse or dense mesh. The choice is made in *Preferences>Program settings*: linear interpolation = coarse, cubic interpolation = dense.

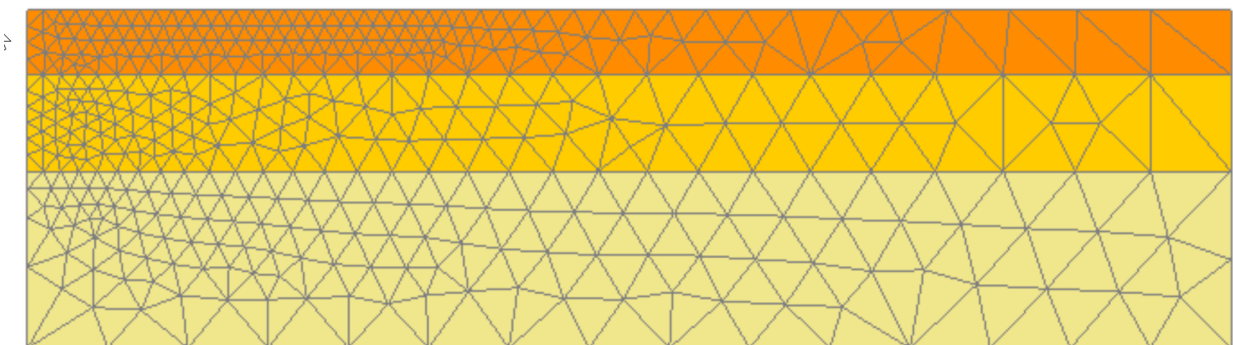


Figure 7: View of the mesh

Bodies definition:

This step is optional but it facilitates the properties and loading assignments for a group of elements.

1. On the **Selection Toolbar**, activate only the option **Select bodies** .
2. Activate  **Body properties**.
3. Right-click on the body corresponding to the layer 1. Enter *Layer1* as name. Specify a colour. Click on **Apply**.
4. Repeat the operation for the *Layer2* and *Layer3*.

3. SHORT-TERM AND LONG-TERM SETTLEMENT ANALYSIS

3.1. Short-term analysis

3.1.1 Model definition

1. Right click on **STATICS** in the **Study** tree view on the right side of the working window.
2. Click on **Add a model**.
3. A new toolbox is open for definition of the Model:
 - Enter **Short term** as "Model name",
 - Tick **Plane strain** as model configuration and **Initial parameters** as type of calculation,
 - Click on **Validate**.

3.1.2 Material properties for the solid elements

Material properties for the solid bodies:

1. Go to the tab **PROPERTIES**.
2. Click on  **Properties for surface bodies**
3. Give a name for the properties set name (*Layer 1-U* for example).
 - In **Elasticity parameters**, choose "Linear isotropic elasticity (+)". Fill in the *Density*, *Young modulus* and *Poisson's ratio* values as defined in the following. Tick **Active** for the *Undrained behaviour*. Set the *Fluid bulk modulus* and the *Porosity*.
 - Click on **Validate**.
4. Repeat previous operations for setting the parameters of *Layer 2-U*.
5. Add a **Properties set** (). Give a name for the properties set name (*Layer 3-U* for example).
 - Fill in the **Elasticity parameters** and undrained behaviour,
 - In **Plasticity parameters**, choose "Mohr-Coulomb (+)" as *Plasticity's criterion*. Fill in the cohesion and friction angle.
 - Click on **Validate**.
6. **Validate** and **Close**

	Density	Young's modulus	Poisson's ratio	Bulk modulus	Porosity	Plasticity
	ρ	E	ν	Kw	n	c ϕ
	kg/m ³	MPa	-	MPa	-	MPa °
Layer 3-U	900	5	0,35		0,35	0,008 0
Layer 2-U	1 100	10,8	0,25	2 000	0,4	
Layer 1-U	1 350	50	0,3		0,3	

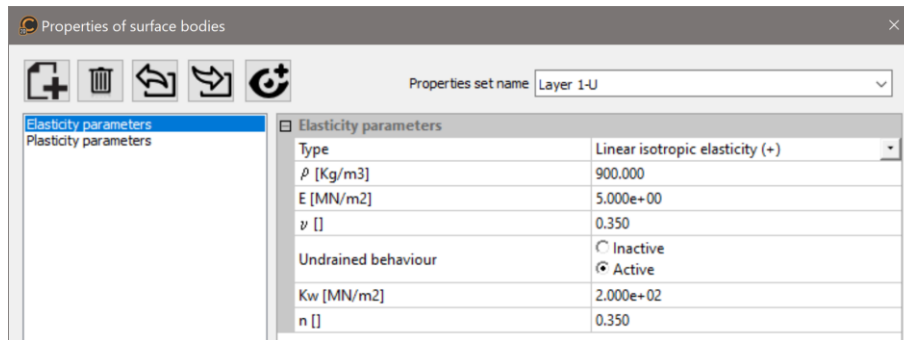


Figure 8: Properties of coupled parameters for undrained behaviour

Assignment of material properties:

1. Click on  **Assign properties** tool.
2. On the left side, a new window is displayed. Click on  **Properties for surface bodies**.
 - Select the surface body corresponding to the *Layer1*.
 - Select the *Layer 1-U* properties set in the list.
 - **Apply**.
3. Repeat the operation for each soil layer.

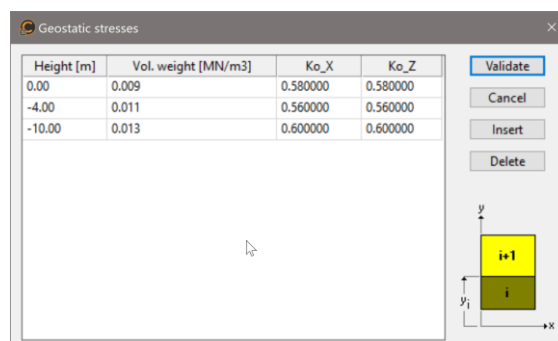
3.1.3 Initial conditions

The initial stress field is initialized as an existing geostatic stress field; vertical stress is linked to the horizontal stress by K_0 value.

The water table being at $y=0$ m, all the layers are saturated with water. We will work with reduced unit weights to define effective stresses.

1. Go to the **INITIAL PARAMETERS** tab.
2. Select  **Geostatic stresses**.
3. Click on **Insert** to define a new layer.
4. Enter the following values then **Validate**.

Height (m)	Unit weight (MN/m ³)	K ₀ _X	K ₀ _Z
0	0.009	0,58	0,58
-4	0.011	0,56	0,56
-10	0.0135	0,60	0,60



3.1.4 Boundary conditions

1. Go to the tab **BOUNDARY CONDITIONS**.
2. On the toolbar, activate  to define side and bottom supports.
3. **Apply**. Supports are automatically affected to the limits of the mesh.



Optional. It is possible to modify the default name assigned to the boundary condition, BCSet1. Press [F2]; enter **Standard fixities** for example.

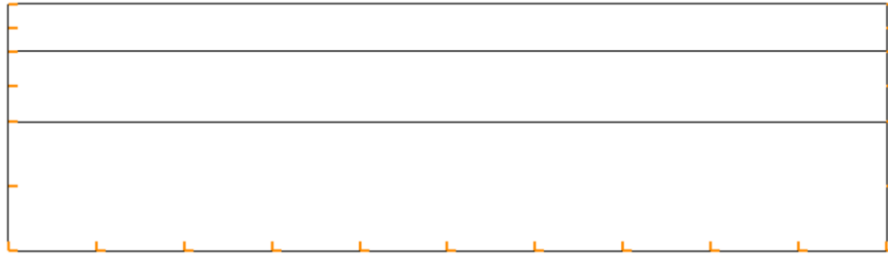


Figure 9: View of mechanical boundaries

4. Go to the **LOADS** tab.
5. On the toolbar, activate  Linearly distributed pressure.
6. Select the segment A.
 - Tick the box "Uniform pressure".
 - Enter the value of $P = 40 \text{ kN/m}^2$.
 - **Apply**.
7. Select the segment B.
 - Untick the box "Uniform pressure".
 - Enter the value of $P_1 = 0 \text{ kN/m}^2$ and $P_2 = 40 \text{ kN/m}^2$.
 - **Apply**.

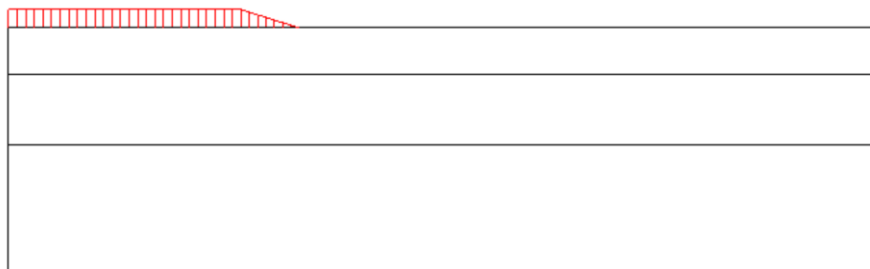


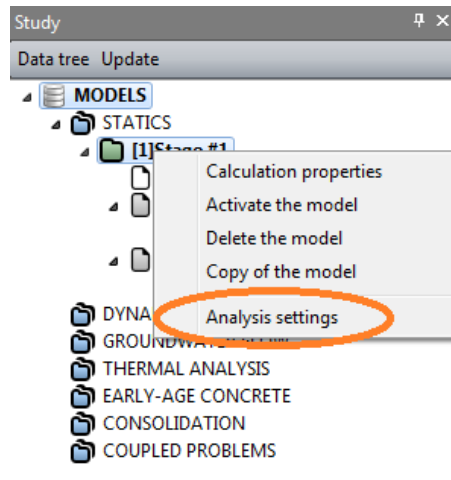
Figure 10: View of the distributed pressure representing the geometry and self-weight of the embankment



Optional. It is possible to modify the default name assigned to the load set, LoadSet1. Press [F2]; enter **Embankment pressure** for example.

3.1.5 Analysis settings

1. Activate the **ANALYSIS** tab.



2. In the STUDY tree, right click on the model title. In the displayed list, select **Analysis settings**.
3. In the **General parameter** section, enter the following values:
 - Iteration process:
 - Max number of increments: 1
 - Max number of iterations per increment: 500
 - Tolerance: 0,01
 - Solution method: 1 – Method of initial stresses
 - Algorithm type: Pardiso
 - Analysis type: Standard
 - Storage: activate storage of total and plastic strains.
4. **Validate.**

3.2. Long-term analysis

This analysis only differs from the previous one by the different properties set assigned to the soils. As we are now performing a drained analysis, the undrained behaviour is no longer activated.

3.2.1 Model definition

1. Right click on model **Short term** in the **Models** tree view.
2. Click on **Copy of the model**.
3. A new toolbox is open for definition of the Model:
 - Enter **Long term** as "Model name",
 - Other model properties remain,
 - Click on **Validate**.
4. The software proposes to share options. This means that a modification in one of the proposed sets (Properties, Boundary conditions or Loadings) will automatically be affected to the other model. Here, we share *Boundary conditions* and *Loadings*.
 - Tick *Boundary conditions* and *Loadings*
 - Click on **Validate**.

3.2.2 Material properties for the solid elements

Material properties for the solid bodies:

1. Go to the tab **PROPERTIES**.
2. Click on  **Properties for surface bodies**
3. Select the set *Layer 1-U*. Click on  **Create properties set**. It copies the set.
4. Give a name for the properties set name (*Layer 1-D* for example).
 - In **Elasticity parameters**, tick **Inactive** for the *Undrained behaviour*.
 - **Validate**.
5. Repeat previous operations for *Layer 2-U* and *Layer 3-U*, renamed in *Layer 2-D* and *Layer 3-D*.
6. **Validate** and **Close**

Assignment of material properties:

1. Use  **Assign properties** tool, to affect the new sets of properties to the appropriate surface bodies.

3.2.3 Initial conditions

No modification.

3.2.4 Boundary conditions

No modification.

3.2.5 Loading

No modification.

3.2.6 Analysis settings

No modification.

4. CONSOLIDATION ANALYSIS

4.1. Model settings

4.2. Material properties for the solid elements

In the consolidation context of CESAR, the Darcy's law for the groundwater flow is written as:

$$v = -k \text{ grad } h$$

where the fluid velocity v is in m/s while the hydraulic head h is in m. It follows that the permeability k is in m/s.

Material properties for the solid bodies:

We complete the data sets of existing materials *Layer 1-D*, *Layer 2-D* and *Layer 3-D* with properties required for the consolidation analysis (flow parameters and coupled parameters).

1. Go to the tab **PROPERTIES**.
2. Click on  **Properties of surface bodies**.
3. Select *Layer 1-D* properties set:
 - In **Flow parameters**, choose "Flow in saturated porous medium". Fill in the permeability values, as defined in the table below.
 - In **Coupled parameters**, choose "Consolidation". Fill in the porosity values, as defined in the following Table. Other values are related to pore fluid properties. The values remain the same for each soil layer as follow: unit weight of pore fluid = **10 kN/m³**; Compressibility of pore fluid = **5 × 10⁻¹⁰ Pa⁻¹**.
 - Click on **Validate**.
4. Repeat the operation to complete the properties set of *Layer 2-D* and *Layer 3-D*.

	Density	Young's modulus (drained)	Poisson's ratio (drained)	Cohesion (kPa)	Friction angle (°)	Dilatancy angle (°)	Porosity	Permeability	
	ρ^*	E	ν	c	φ	ψ	n	k_{xs}	k_{ys}
	kg/m ³	MPa	-				-	m s ⁻¹	
<i>Layer 3</i>	900	5	0,35	8	0	0	0,35	1,2. 10 ⁻⁷	1,2. 10 ⁻⁸
<i>Layer 2</i>	1100	10,8	0,25				0,4	10 ⁻⁶	10 ⁻⁷
<i>Layer 1</i>	1350	50	0,3				0,3	5,8. 10 ⁻⁶	5,8. 10 ⁻⁶



The compressibility of the pore fluid C_w is the inverse of the fluid bulk modulus K .

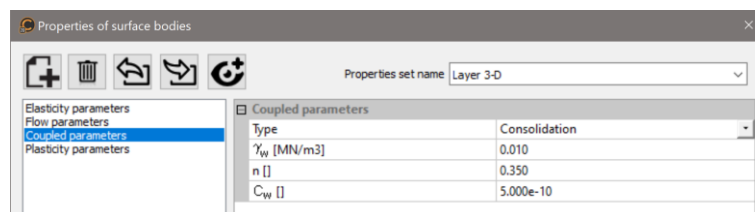


Figure 11: Parameters γ_w and C_w of the pore fluid remain unchanged in each layer

4.3. Initial condition

The initial stress field is initialized as an existing geostatic stress field; vertical stress is linked to the horizontal stress by K_0 value.

1. Go to the **INITIAL PARAMETERS** tab.
2. Activate the toolbar initialization by **Stress** 
3. Select  **Geostatic stresses**.
4. Click on **Insert** to define a new layer.
5. Enter the following values then **Validate**.

Height (m)	Unit weight (MN/m ³)	K ₀ _X	K ₀ _Z
0	0.009	0,58	0,58
-4	0.011	0,56	0,56
-10	0.0135	0,60	0,60



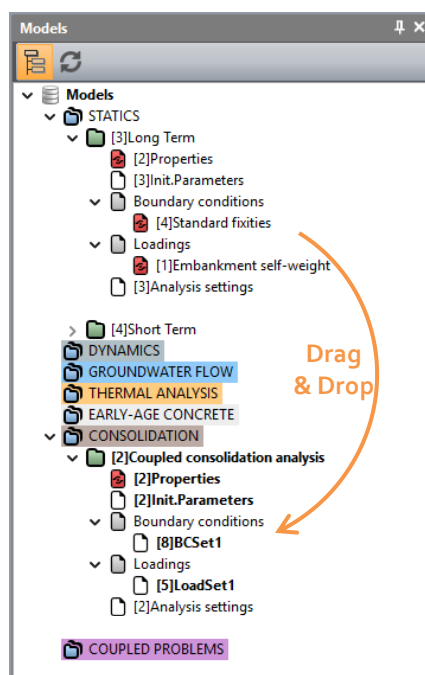
In elastic linear analysis, the initial stress field has no influence on the final settlement.

4.4. Boundary conditions

In coupled analysis, there are 2 kinds of boundary conditions: "Mechanical" and "Groundwater flow".

First, we can copy the "Mechanical" boundary conditions from the previous models by drag-and-drop action.

1. Select the set **Standard fixities** of model "**Long term**".
2. Drag and drop on the BCSet1 of this model.
3. Accept to change the name.



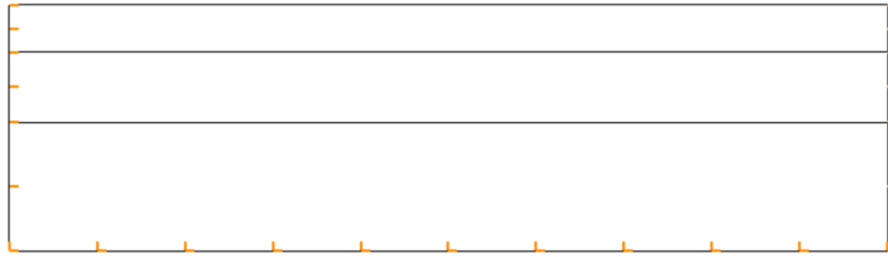


Figure 12: View of mechanical boundary conditions

4. On the model tree, right-click on "Boundary conditions" and Click "Add a boundary conditions set". Name it *Hydraulic BC*.
5. Activate the toolbar of hydraulic boundary $\bar{H}$.
 - On the toolbar, activate $\bar{h}$ to prescribe a constant load.
 - Selection the segments ABC, the top surface of the model.
 - Fill in the value **0 m** on the dialog box.
 - **Apply**.

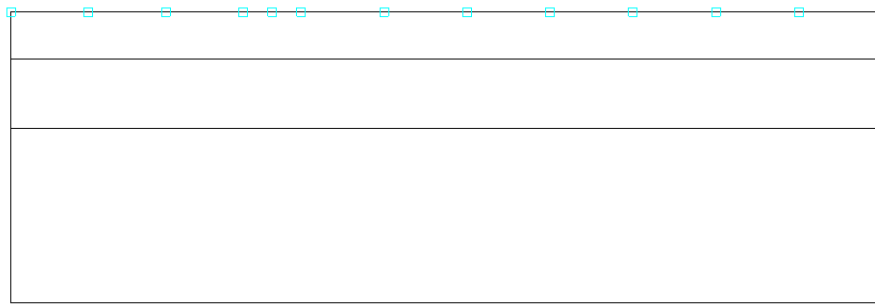
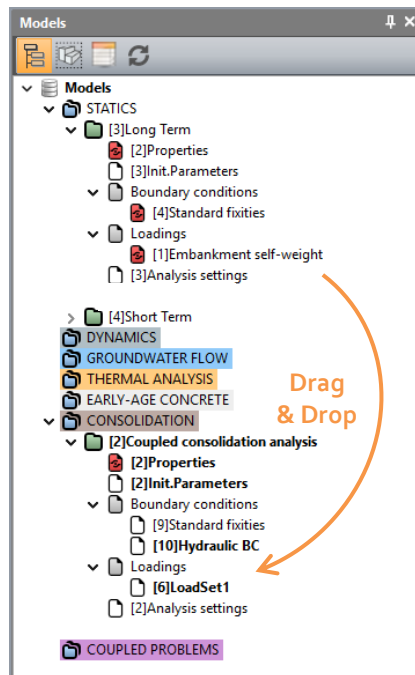


Figure 13: View of hydraulic boundary conditions

4.5. Loading

Alike the boundary conditions, we can copy the load set from the previous models by drag-and-drop action.

1. Select the set **Embankment self-weight** of model "Long term".
2. Drag and drop on the LoadSet1 of this model.
3. Accept to modify the name.

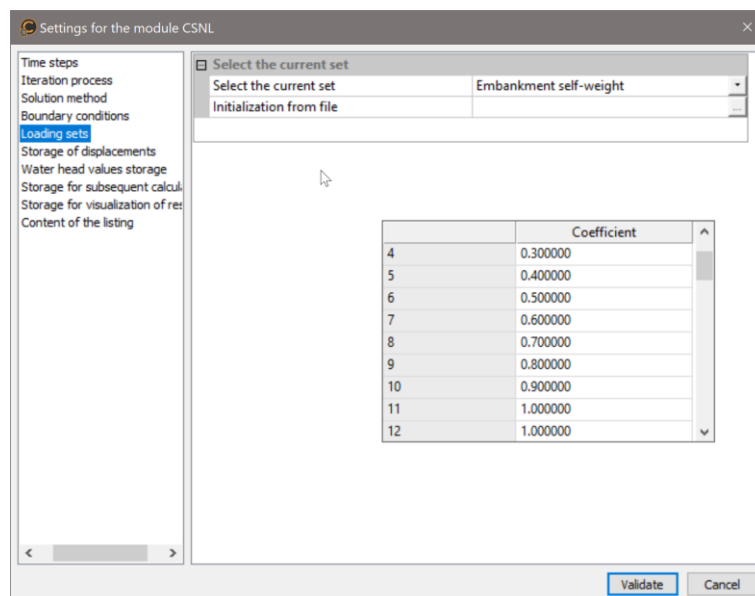


4.5.2 Analysis settings

1. Go to the tab **ANALYSIS**.
2. Click on  **Analysis settings**.
3. In the **Time steps** section, enter the following values:
 - Number of time steps: 51
 - Start time value at 0.0 second
 - Activate **By groups** to define variation of time steps loading
 - Define 3 groups of time steps loading as follow:

	Number of steps	Steps value
1	10	5 000
2	10	50 000
3	30	500 000

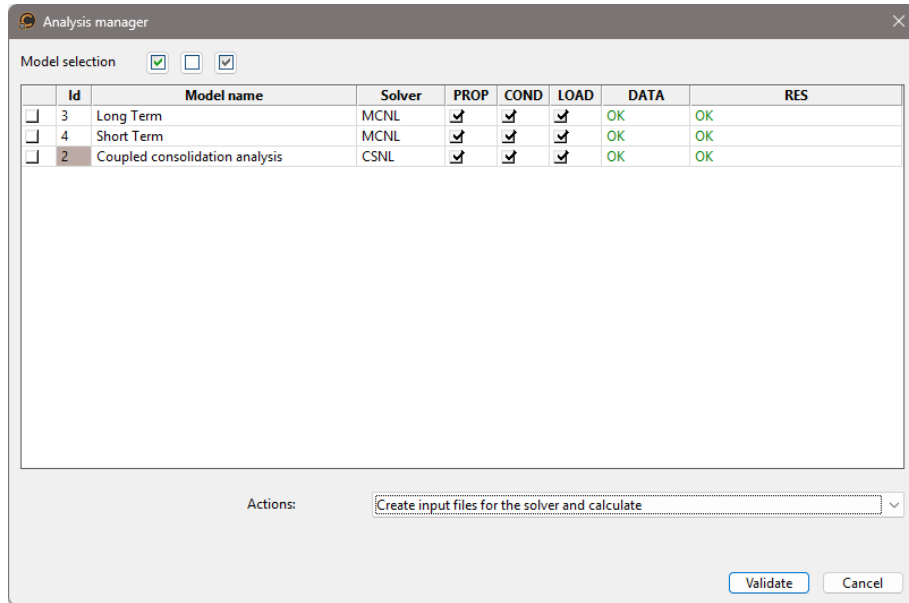
4. In the **Iteration process** section, enter the following values:
 - Max number of iterations: 500
 - Tolerance: 0.001
5. In the **Boundary conditions** section, set all coefficient and relaxation time functions to 1.00 (default option). This is to control that all displacements, temperature and pressure boundary prescriptions remain constant during all steps of calculation.
6. In the **Loading sets** section, set the coefficient load functions as an incremental function as the following figure.



7. If user needs to perform uncoupled hydro-mechanical analysis or a multiple staged construction analysis, user can select the options *Storage of displacements*, *Storage of water head* and *Storage for subsequent calculation*.
8. In the **Storage for visualization of results** section, user can here select the time steps for which he requires results.
 - Click "Storage for all time steps" to store the results for all the time steps.
 - Tick "Primary and secondary results". Secondary results include the stress field.
9. User can also choose by himself the results that will be written in the listing file (.list file) in the **Content of the listing** section.
10. **Validate**.

5. SOLVE

1. Go to the **ANALYSIS** tab.
2. Click on  **Analysis manager**.
 - Select the 3 models.
 - Select **Create input files for the solver and calculate**.
 - Click on **Validate**.



 The iteration process is displayed on the **Working window**. It ends with the message "End of analysis in EXEC mode".

 CESAR-LCPC detects if the models are ready for calculation. All steps should be validated with a tick mark.

6. RESULTS

6.1. Comparison of settlements

We can compare the settlements reached with short-term and long-term analyses with those obtained at the start and end of the consolidation process.

1. Go to tab **RESULTS**.
2. On the toolbox **Results selection**, select the **Time step [51]**
3. Click on  **Type of results to display**.
 - Select **Deformed as Mesh**,
 - Tick the box "Active" and select **Total displacement** in *Contour plots* section,
 - **Apply**.
4. Click on  **Displacement settings**.
 - Select **Manual** as *Scale*,
 - Set that a value of 2 mm is represented by 0.1 m,
 - **Apply**.

At the bottom of the working window, the maximum of total displacement is displayed.

Analysis		u
Short term	5 th increment (100% of the load)	31 mm
Long term	5 th increment (100% of the load)	71 mm
Consolidation	11 th time step (100% of the load)	32 mm
	51 st time step	73 mm

We show here how the various analyses get close results for the prediction of the settlements.

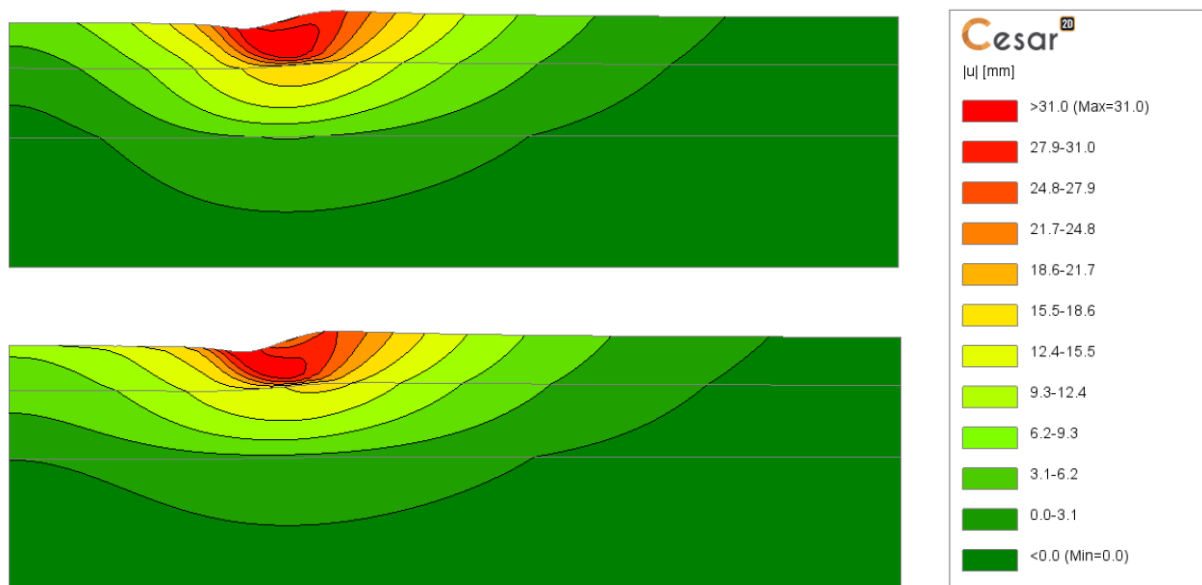


Figure 14: View of the total displacements at short term (time steps #6)

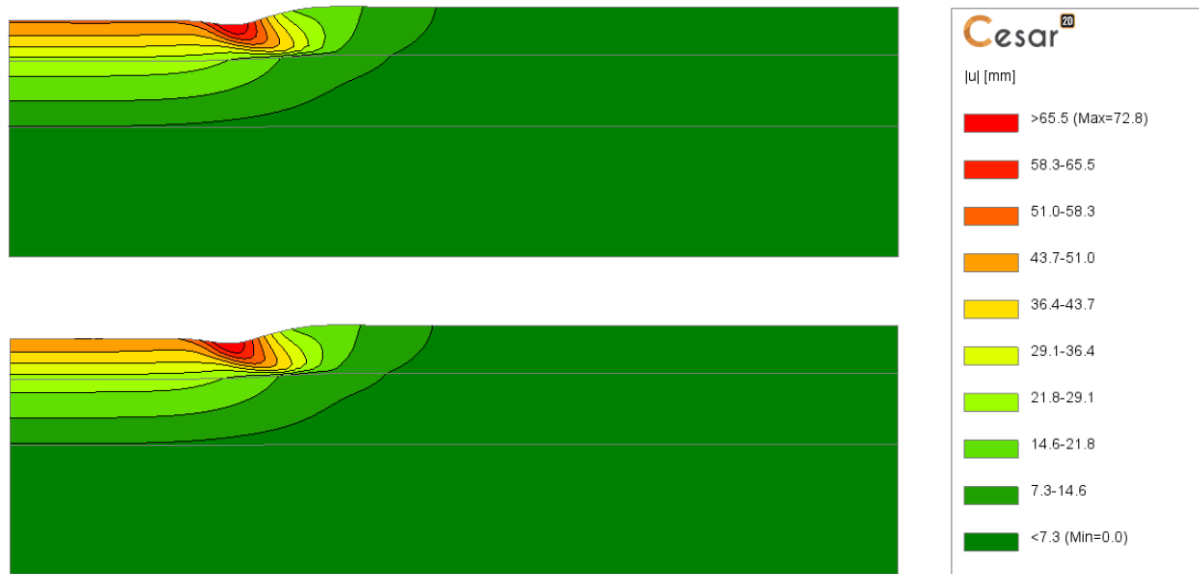


Figure 15: View of the total displacements at long term (time steps #51)

6.2. Other display of results for the consolidation model

1. Go to tab **RESULTS**.
2. On the toolbox **Results selection**, select the **Time step [51]**
3. Click on  **Type of results to display**.
 - Select **Deformed** as *Mesh*,
 - Tick the box "Active" and select **Hydraulic pressure** in *Contour plots* section,
 - **Apply**.
4. Click on  **Contour plot settings**.
 - Tick the box "Activate" and select **Areas** as *plot style*,
 - Tick the box "Contour lines" and select **Grey** as *Color set*,
 - Select **Groundwater** as *Color scheme*,
 - **Apply**.
5. Click on  **Legend**.
 - Select **Contour plot** as *Legend type*,
 - Tick the box "Legend border",
 - **Apply**.

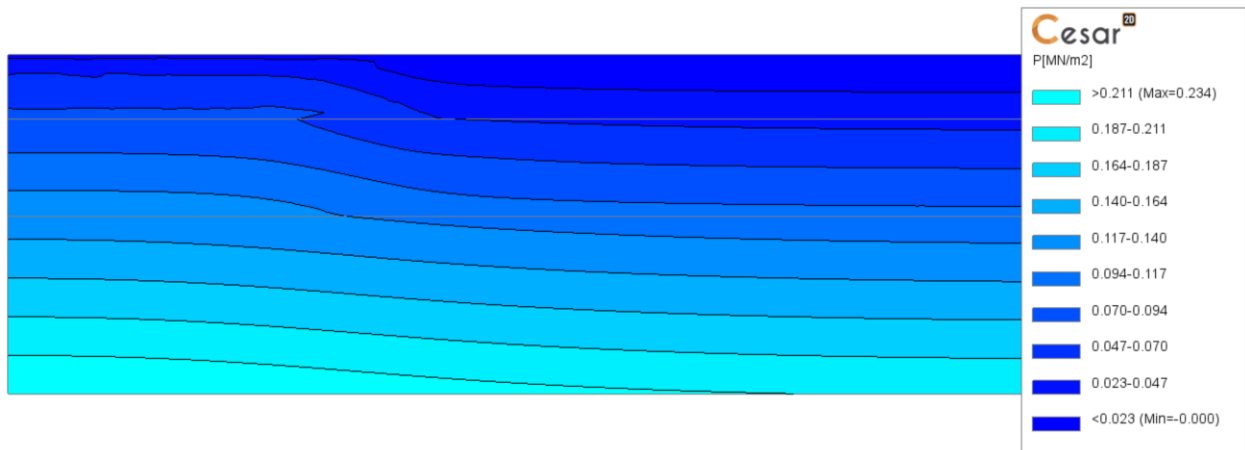


Figure 16: Consolidation - view of the hydraulic pressure field at time steps [6]

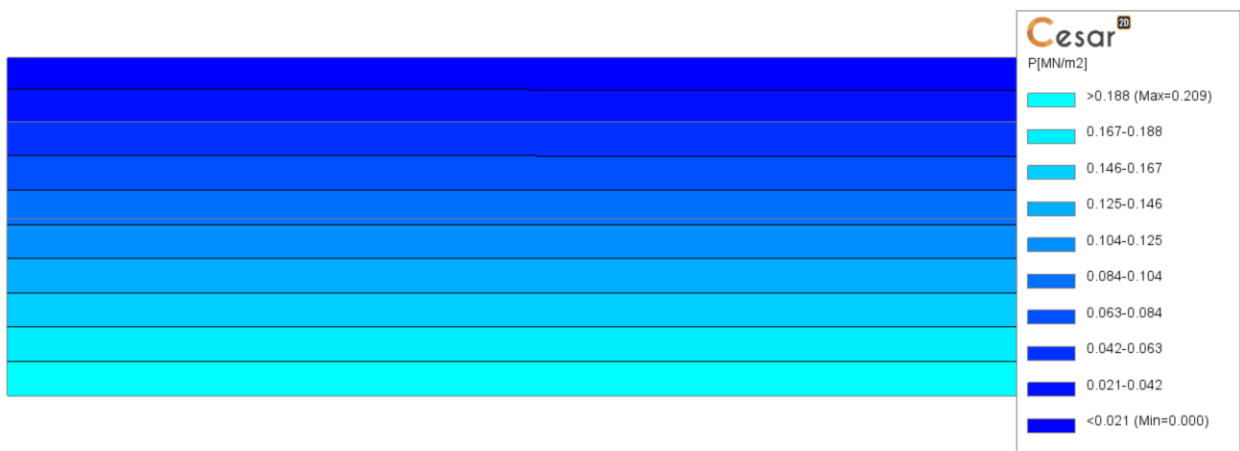


Figure 17: Consolidation - view of the hydraulic pressure field at time steps [51]

Click on  **Animation.**

- Select **All entities** as *Animation type*,
- Click  backward to the initial time step,
- Untick the box "Repetition of the animation",
- Click  to play the animation. The animation shows the evolution of vertical displacement with respect to the time step loading function.

6.3. Graphs

6.3.1 Water head level in mid layer

1. Go to tab **CHARTS**.
2. Activate ☐ **Points** on the **Entities for charts** toolbar.
 - Create 4 observed points as follow:

Name	Px [m]	Py [m]	Representation of point
P0	0	0	Center of embankment line
P1	25	0	Edge of embankment line
P2	20	-2	Mid position of layer 1
P3	10	-7	Mid position of layer 2

3. Activate  **Set of points** on the Entities for charts toolbar.
 - Activate only the toolbar  **Select vertices**.
 - Click on points "P2" and "P3", enter a name "Midpoints" in the *Set of points* box, then click on **Add**.
4. Activate  **Charts for a set of points** on the **Charts** toolbar.
 - Select "Water head" as Y-Axis,
 - Select "Midpoints" as *Set of points*,
 - Click on **Apply**.

 Water head at the embankment points is null due to the boundary conditions.

 User can set the display colour and the type of marker on the **Chart settings** . If user would like to draw a graph by himself, user can export the data with the **Exports of charts data** .

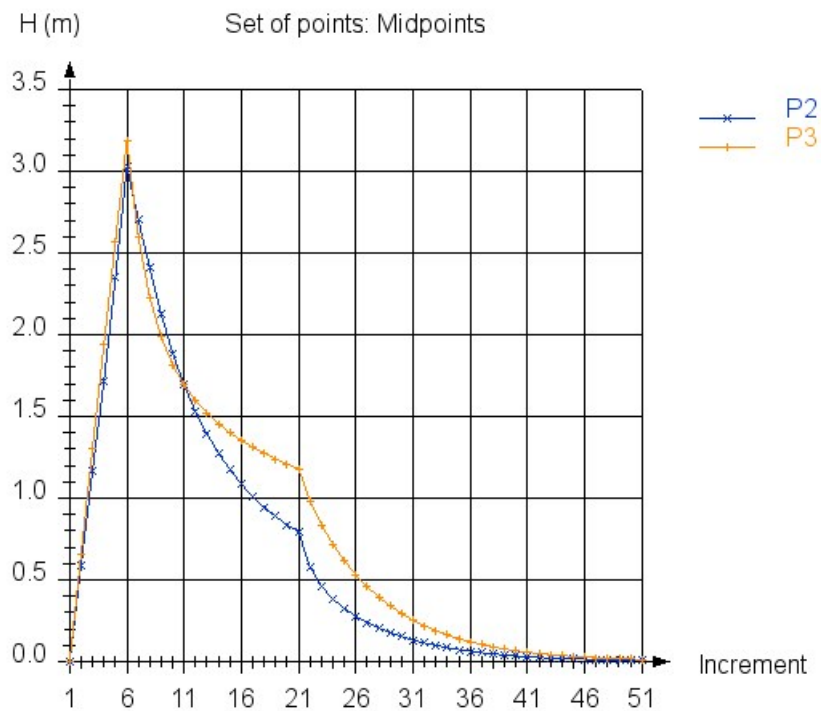


Figure 18: Evolution of water head in the midpoints over the incremental time steps

6.3.2 Vertical displacement on the embankment line

1. Go to tab **CHARTS**.
2. Activate  **Lines** on the **Entities for charts** toolbar.
 - Click on points "Po" and "P1" then double click until a red line appears.
3. Activate  **Line set** on the **Entities for charts** toolbar.
 - Click on the red line connecting Po and P1,
 - Enter a name (for example "Embankment") in the *Line set* box,
 - Click on **Add**.
4. Activate  **Charts for a line set** on the **Charts** toolbar.
 - Select "Water head" as *Parameter*,
 - Select the cut line of centre axis as *Line set*,
 - Select the following time steps to better observe the evolution of displacement, then click on **Apply**.

N° of time step	Representation of loading		
	Initial stress	Embankment pressure	Time
Time step_0	Geostatic stress	-	-
Time step_1		8 kN/m	5 000 s
Time step_5		40 kN/m	25 000 s
Time step_50		40 kN/m	end (around 232 days)

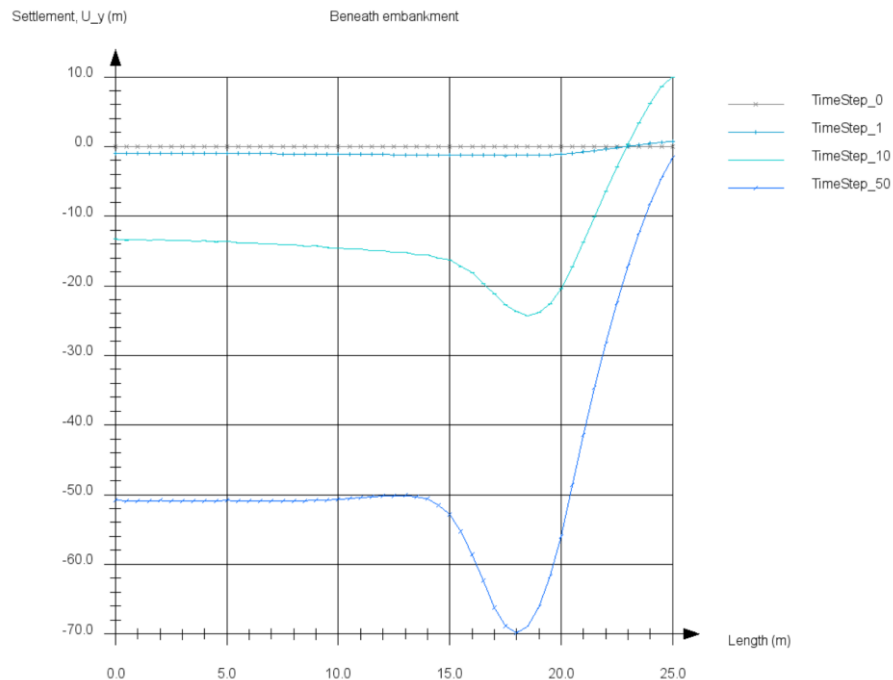


Figure 19: Evolution of vertical displacement below the embankment with time

Edited by :



8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tél. : +33 1 49 76 12 59

cesar-lcpc@itech-soft.com

www.cesar-lcpc.com

© itech - 2025